DESIGN AND SIMULATION OF MATERIALS

IN COMSOL MULTIPHYSICS SOFTWARE

SUMMER PROJECT REPORT

Submitted by

SAIMOHAN R (2021105328)

in partial fulfillment for the award of the degree of

BACHELOR OF ENGINEERING

in

ELECTRONICS AND COMMUNICATION ENGINEERING

COLLEGE OF ENGINEERING, GUINDY

ANNA UNIVERSITY: CHENNAI 600 025

JULY - NOVEMBER 2023

ANNA UNIVERSITY: CHENNAI 600 025

BONAFIDE CERTIFICATE

Certified that the summer project report " **C0MSOL MULTIPHYSICS SOFTWARE**" is the bonafide work of " **SAIMOHAN R (2021105328)** ", who carried out the project work for EC-5512, 5th semester, Summer Project in the duration July - November 2023 under my supervision. Certified further that to the best of my knowledge the work reported here in does not form part of any other thesis or dissertation on the basis of which a degree or award was conferred on an earlier occasion on this or any other candidate

SIGNATURE	**SIGNATURE**
Dr.M.Meenakshi	**Dr.S.Shanmugapriya**
HEAD OF THE DEPARTMENT	**SUPERVISOR**
Professor	Associative Professor
Department of ECE	Department of ECE
College of Engineering Guindy	College of Engineering Guindy
Anna University	Anna University
Chennai-600025	Chennai-600025

ACKNOWLEDGEMENT

I express my sincere gratitude to Dean, **Dr.L.Suganthi**, Professor, College of Engineering, Guindy for her support throughout the project.

I expresss my sincere gratitude to my Head of the Department **Dr.M.Meenakshi**, Professor, Department of Electronics & Communication Engineering for her enthusiastic encouragement and support throughout the project.

I present my sincere thanks and gratitude to my project supervisor and co-ordinator, **Dr.S.Shanmugapriya** , Professor, Department of Electronics and Communication Engineering for her whole hearted support, patience, valuable guidance, technical expertise and encouragement in my project.

I thank all the teaching and non-teaching staff of the Department of Electronics and Communication Engineering, for their kind help and co-operation during the course of my project.

ABSTRACT

COMSOL Multiphysics is a powerful and versatile software package designedfor simulating and solving complex multiphysics phenomena across various scientific and engineering disciplines. This software enables researchers, engineers, and scientists to model, analyze, and optimize systems that involve the coupling of multiple physical phenomena, such as heat transfer, fluid flow,structural mechanics, electromagnetics, and chemical reactions, among others.

Key features of COMSOL Multiphysics include a user-friendly graphical interface, a wide range of predefined physics modules, customizable modelingcapabilities, and the ability to define and solve coupled physical equations.
Users can create intricate simulations, visualize results, and gain insights intothe behavior

TABLE OF CONTENTS

CHAPTER 1

BASICS OF COMSOL SOFTWARE

1.1 BASICS OF COMSOL SOFTWARE

- COMSOL Multiphysics is a software package that can be used to create and solve models involving different physics domains, such as mechanics, acoustics, electromagnetics, fluid dynamics, heat transfer, andmore. It can be applied to various fields of engineering, science, and industry.
- COMSOL Multiphysics is a software that allows you to create and solvemodels involving different physics domains, such as mechanics, acoustics, electromagnetics, fluid dynamics, heat transfer, and more .
- You can also couple related physical applications together to include allthe necessary factors for a complete model.
- The software uses the finite element method to solve systems of partialdifferential equations that describe the physics of your model.
- To use COMSOL Multiphysics, you need to follow these basic steps:
- - Define the geometry of your model
- - Choose the physics interfaces and equations
- - Specify the material properties and boundary conditions
- - Mesh the geometry and solve the model
- - Analyze and visualize the results

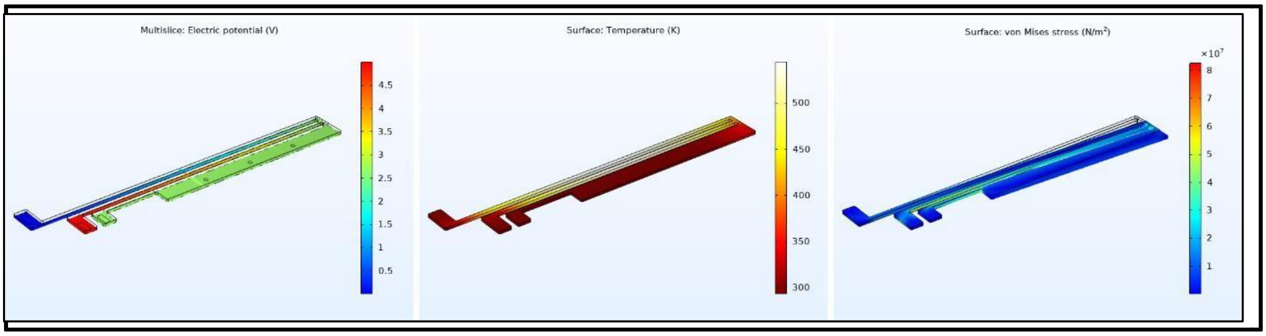

Fig1.1 Basic 2D model

1.2 Defining Multiphysics Models Automatically with Multiphysics Interfaces.

- Introduce the automatic approach
- Discuss why and how this approach is advantageous
- Show where to find the lists of predefined multiphysics interfaces
- Outline the procedure for implementing the approach
- Introduce the example model used throughout the course and demonstrate use of the automatic approach during the model buildingprocess

1.3**Defining Multiphysics Models Manually with PredefinedCouplings**

- Introduce the manual with predefined couplings approach
- Discuss use cases for the approach and how it is advantageous
- Outline the procedure for implementing the approach
- Demonstrate use of the approach using the example introduced in part 1

1.4 **Defining Multiphysics Models Manually with User-defined Couplings**

- Introduce the manual with user-defined couplings approach
- Discuss use cases for the approach
- Outline the procedure for implementing the approach and explain thelogic behind it
- Demonstrate use of the approach using the example introduced in part 1

1.5 **Viewing and Accessing Equations and Variables forPhysics Feature Nodes**

- Show the multiple ways you can see the equations for the physics of amodel in the software
- Show the multiple ways you can see the variables for a physics node andhow they are defined
- Discuss the advantages of using equation view nodes and thereport node for accessing physics equations and variables
- Demonstrate creating a user-defined multiphysics coupling

1.6 **Coupling Physics Between Model Components forMultiphysics Models**

- Introduce a multicomponent implementation of the manual approachwith user-defined couplings
- Discuss use cases and examples for these type of couplings
- Demonstrate creating a user-defined multiphysics coupling betweenmultiple model components

Fig1.2 Builder options used in COMSOL

1.7 PARAMETERS:

Name	Expression	Value	Description
w_core	460[nm]	4.6E-7 m	width of core
h_core	200[nm]	2E-7 m	heigth of core
w_clad	1000[nm]	1E-6 m	width of cladding
h_clad	1000[nm]	1E-6 m	heigth of cladding
n_core	3.48	3.48	refractive index of core
n_clad_over	1.51	1.51	refractive index of claddi...
n_clad_un...	1.44	1.44	refractive index of claddi...
lda0	1.55[um]	1.55E-6 m	operating wavelength
f0	c_const/lda0	1.9341E14 1/s	operating frequency

Fig 1.3 Parameters used to denote geometry of a material

1.8 MATERIALS

Materials

Label: Materials

Material Overview

Material	Selection
Si (mat1)	Domain 4
siON (mat2)	Domains 2–3, 5
siO2 (mat3)	Domain 1

Fig 1.4 materials used in COMSOL

- After selecting required parameters and materials mesh process will beintroduced to the particular material (silicon rib).
- After mesh process steady step will be implementedStep1:

  boundary mode analysis

  Step 2: boundary mode analysis 1
  Step 3: frequency domain

MESH OF SILICON RIB

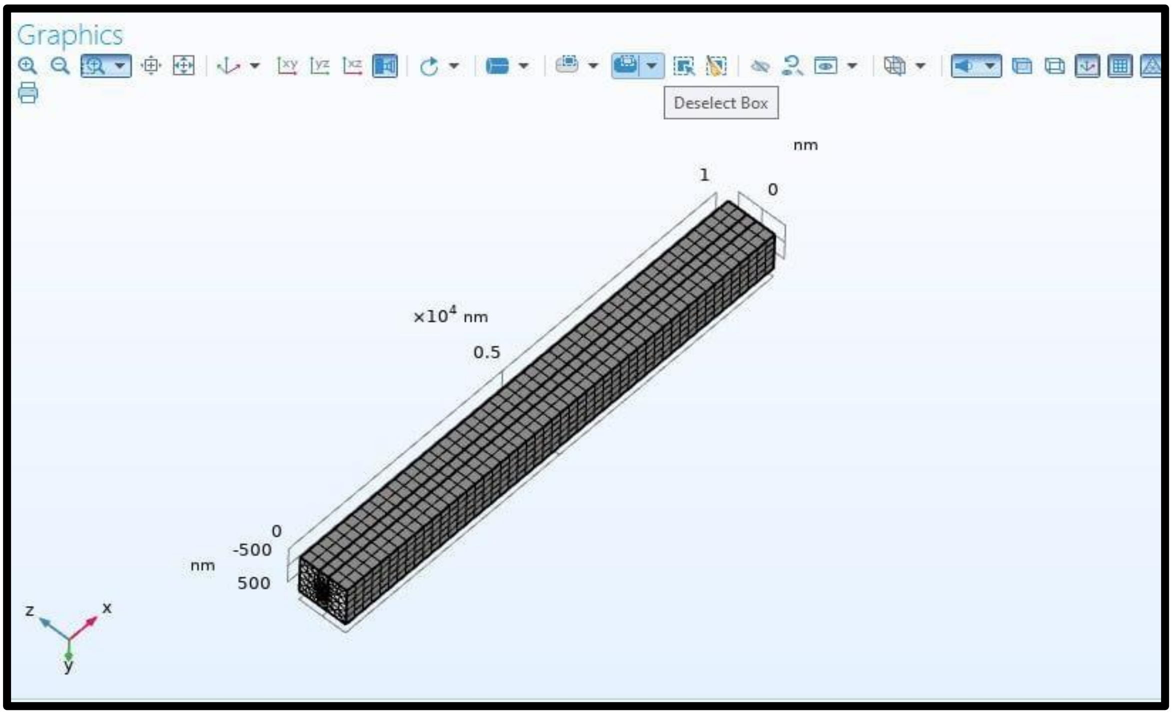

Fig 1.5 mesh analysis

- After the mesh and study process the derived value of the material will be

Frequency	193.41thz
Reflectance	4.7264E-2D
Transmitance	1000
Absorptance	13856E-13

- Final result will be exported in electric field and we can see the flowOf electric field in the material.

RESULT : [Electric field]

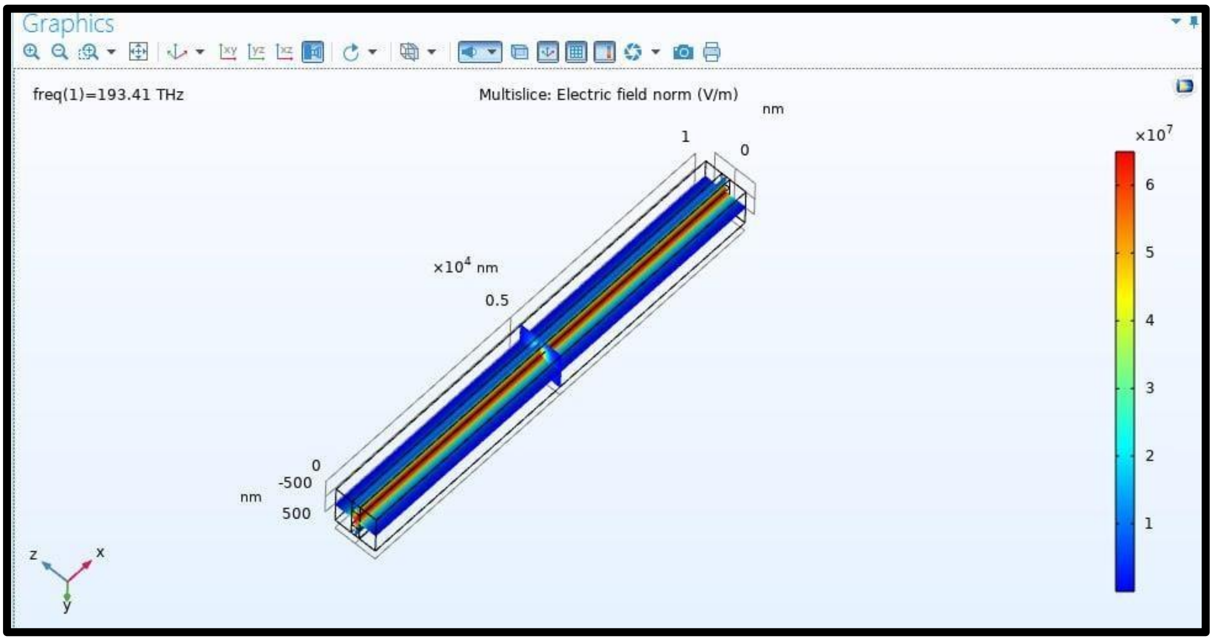

Fig 1.6 2D analysis of electric field

ANIMATED VIEW:

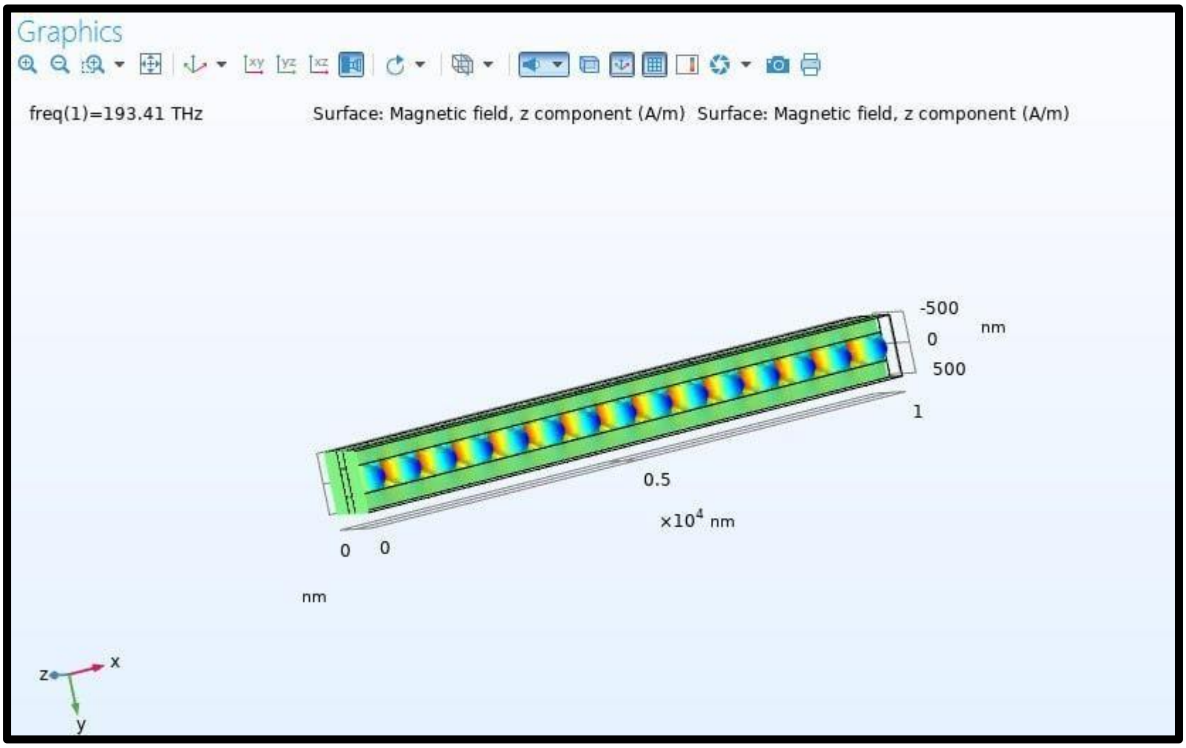

Fig 1.7   3D analysis of magnetic field

CHAPTER 2

DESIGN AND SIMULATION OF RECTANGULAR WAVEGUIDE

2.1 MODE ANALYSIS:

When analyzing an arbitrary 3D waveguide structure, it is important to understand which types of electromagnetic waves are allowed to propagate at a given frequency. These wave regimes are determined by the resonant modes, which can be excited in a 2D transverse cross section of a waveguide.

Such modes can be fully described by the global complex-valued propagation constants and the space distributions of all three components of electric field (so-called mode shape). Transmission regimes in waveguides with a constant cross section can be defined completely based on these electromagnetic characteristics.

We can also use this information for frequency-domain studies of scattering characteristics in more complex structures.

Well-known analytic solutions are only available in the literature for some RF designs, such as coaxial lines and hollow waveguides with rectangular or circular cross sections. For any other configuration with arbitrary shape and material combinations including all typical optical fibers and integrated waveguides it is necessary to use numeric mode analysis shows the formulation of numeric mode analysis in the equation section of the setting window.

To perform a mode analysis, you will need to plug a given frequency into the Helmholtz equation for electric fields and then search for a solution in the form of a wave traveling in the out-of-plane direction. For that purpose, you can use the finite element method (FEM) and an eigenvalue solver.

Mode analysis should not be confused with the more general modal analysis. The latter is referred to as eigen frequency and can be used for finding resonant or natural modes and eigenfrequencies in a system of any possible dimension, including 2D, 2D axisymmetric, and 3D.

You can perform mode analysis in the RF Module or Wave Optics Module, both add-on products to COMSOL Multiphysics, by using the following features: the Electromagnetic wave frequency domain physics interface for 2D or 2D axisymmetric geometry and a mode analysis study.

MODEL INPUTS:

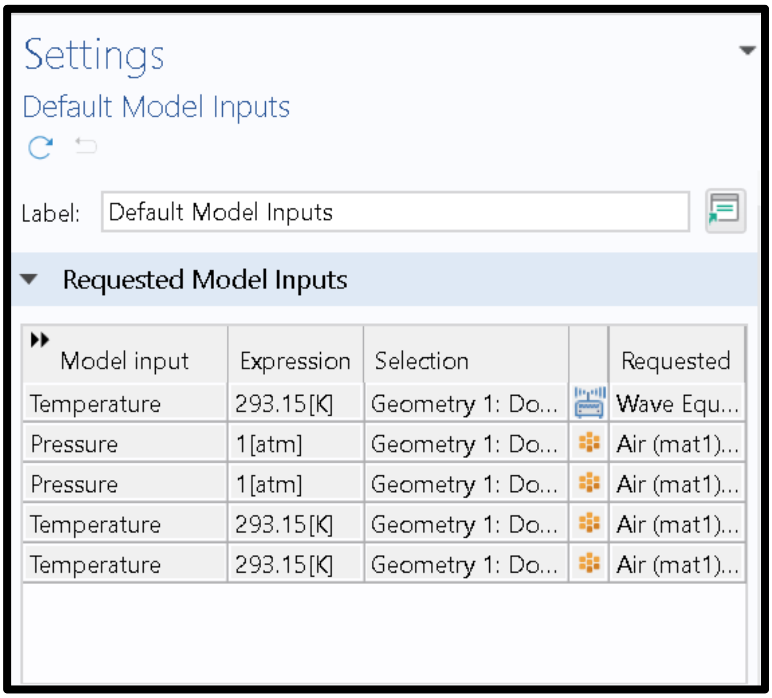

Fig 2.1 input used for designing rectangular waveguide

MATERIALS TO DESIGN RECTANGULAR WAVEGUIDE:

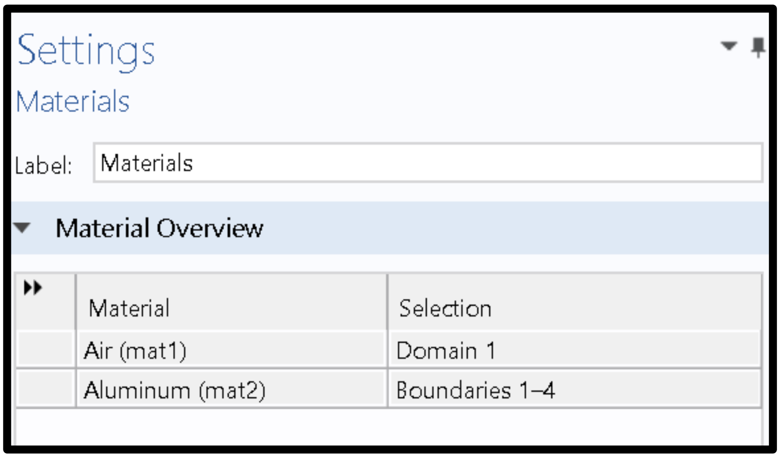

Fig 2.2 material used in rectangular waveguide

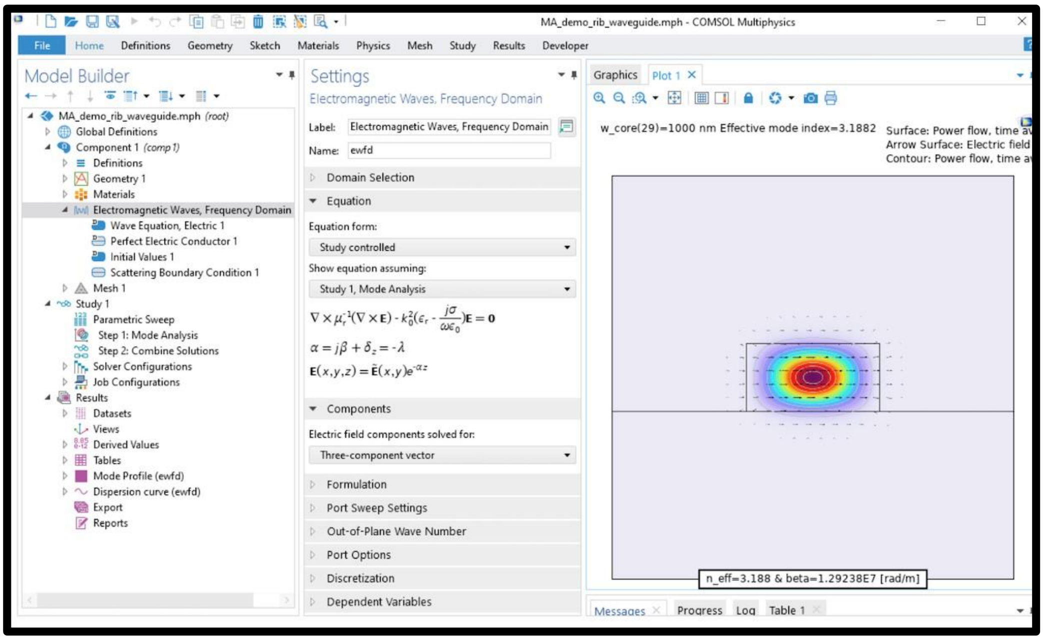

Fig 2.3 EM wave in frequency domain

2.2 PHYSICS SETTINGS:

The goal is to find a wave that is propagating in the out-of-plane direction. To do so using the Electromagnetic waves,frequency domain physics interface in 2D, open the setting window for the physics interface and make sure the Three component vector option is selected in the component section.

Mode analysis is an eigenvalue study, so there is no need to use any source conditions. However, you should still define proper boundary conditions since they will have an affect on mode shape and mode damping and leakage. Note that external boundaries can be metallized or open. If you are working with metallized boundaries, you can use the default perfect electric conductor or impedance boundary condition. In order to describe open boundaries, you can use the scattering boundary condition or a perfectly matched layer.

2.3 MESH AND STUDY SETTING:

A variant of the mode analysis is study settings for electromagnetics problems is shown in Figure 3 below. By default, the effective mode index transform is selected, which is usually the best choice for electromagnetic waves. With such a transform, you can assume that the so-called effective index of mode will be used as a declarative characteristic of the mode.

In the mode analysis frequency field, you should enter the frequency at which you want to find resonant modes. As you can see below, the next study setting listed is mode search method. If manual search is selected here, you should set the initial guess in terms of the effective index of mode in the search for modes around field and the desired number of modes. The solver will search for modes near that guess and return the expected number of different modes if possible. For the region search, you should specify the approximate number of modes and the region of the complex effective index.

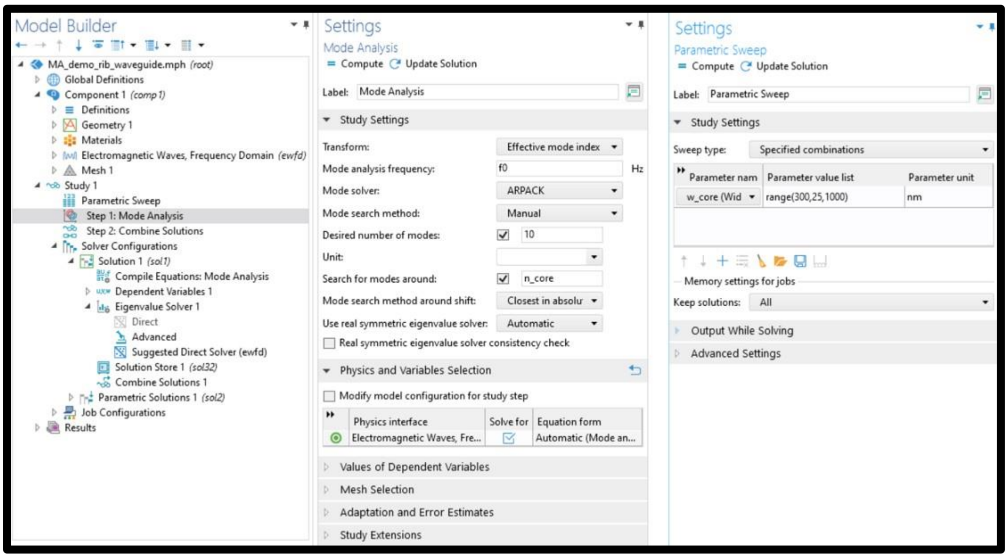

Fig 2.4 mode analysis

MESH PROCESS OF RECTANGULAR WAVEGUIDE:

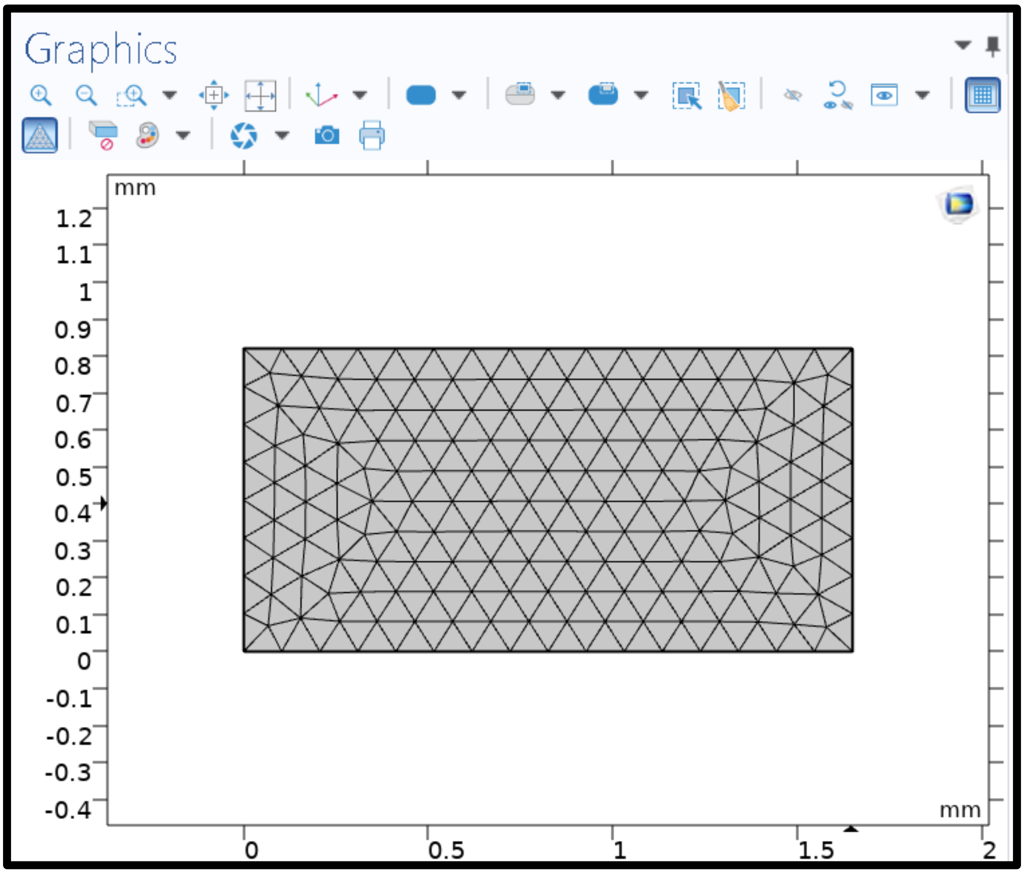

Fig 2.5 Mesh process in rectangular wave guide

After mesh process next stage of study will be applied to the particular material.

In study stage mode analysis and solver configuration will be applied.

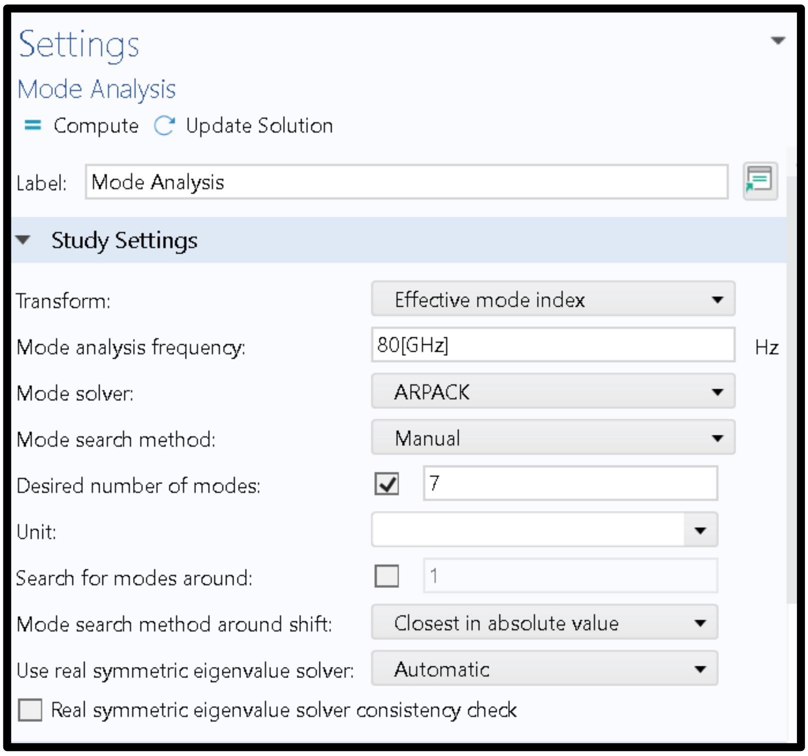

Fig 2.6 solver configuration

The above displayed option should be choosen and process the material for nextstage.

Derived values:

Effective mode index	Complex propagation constant (rad/m)
9.6466E-4-2.3465i	3934.4+1.6174i
9.3957E-4-2.0536i	3443.3+1.5753i
5.3026E-4-2.0386i	3418.1+0.88907i
7.4949E-4-0.53734i	900.94+1.2567i
-7.4949E-4+0.53734i	-900.94-1.2567i
-5.3026E-4+2.0386i	-3418.1-0.88907i
-9.3957E-4+2.0536i	-3443.3-1.5753i

Fig 2.7 Derived values of rectangular waveguide

The above mensioned values are the effective index and propagation constantOf the rectangular wave guide.

Result:

This shows the flow of electric field in the medium

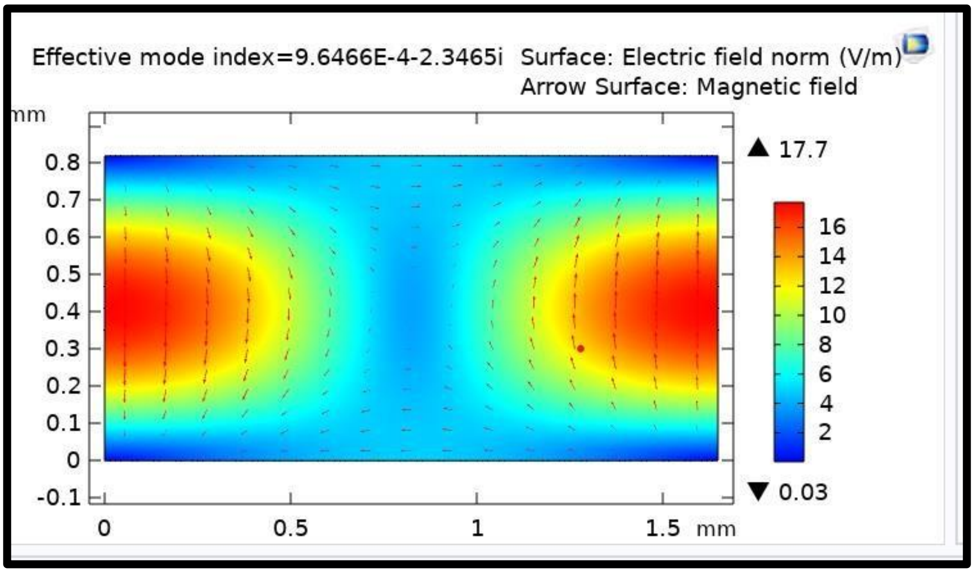

Fig 2.8 3D analysis of electric field

GRAPHICAL REPRESENTATION:

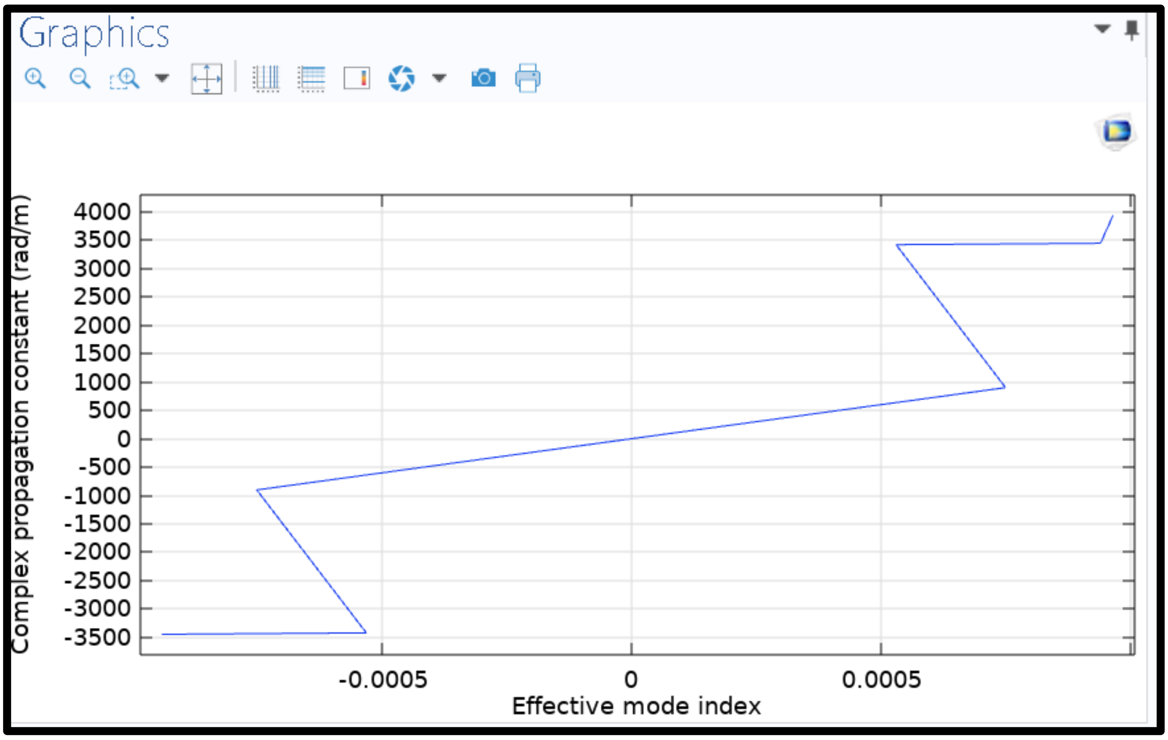

Fig 2.9 Graphical representation

CHAPTER 3
DESIGN AND SIMULATION OF PHOTONIC CRYSTAL

3.1 PHOTONIC CRYSTAL:

- Photonic crystal devices are periodic structures of alternating layers of materials with different refractive indices. Waveguides that are confined inside of a photonic crystal can have very sharp low-loss bends, which may enable an increase in integration density of several orders of magnitude. This is a study of a photonic crystal waveguide.

- The crystal features a grid of GaAs pillars. Depending on the distance between the pillars, waves within a certain frequency range will be reflected instead of propagating through the crystal. This frequency range is called the photonic band gap.

- When some of the GaAs pillars in the crystal structure are removed, a guide for the frequencies within the band gap is created. Light can then propagate along the outlined guide geometry.

- The Wave Optics Module can be combined with any other module to simulate multiphysics phenomena, all of which seamlessly integrate with the core COMSOL Multiphysics software platform. This means that your modeling workflow remains the same, regardless of the application area or physics you are modeling.

- Additionally, you can simulate how various physical phenomena can be used for modulation purposes, such as acousto-optical, electro-optical, and magneto-optical effects.

- By combining with a mass transport simulation, you can compute realistic refractive index profiles with anisotropic diffusion coefficients and use the results in an electromagnetics analysis.

- Photonic crystal fiber (PCF) is also known as micro structured or holey fiber that uses Photonic crystal, which is formed using a periodic array of microscopic air holes that run along the entire fiber length. Contrary to normal fiber optics, PCFs use total internal reflection or light confinement in hollow core methods to propagate light. Light propagation in PCFs is far superior to standard fiber, which uses constant lower refractive index cladding.

- Photonic crystal is a low-loss periodic dielectric medium constructed using a periodic array of microscopic air holes that run along the entire fiber length.

PARAMETERS TO DESIGN PHOTONIC CRYSTAL:

Parameters

Name	Expression	Value	Description
p	1.5[um]	1.5E-6 m	pitch
d1	.15*p	2.25E-7 m	diameter

Fig 3.1 Parameters used in photonic crystal

MODEL INPUT REQUIRED TO DESIGN A MATERIAL:

Requested Model Inputs

Model input	Expressior	Selection		Requestec
Temperature	293.15[K]	Geometry 1: D...		Wave Eq...
Pressure	1[atm]	Geometry 1: D...		Air (mat...
Pressure	1[atm]	Geometry 1: D...		Air (mat...
Temperature	293.15[K]	Geometry 1: D...		Air (mat...
Temperature	293.15[K]	Geometry 1: D...		Air (mat...

Fig 3.2 Input used in photonic crystal

3.2 MATERIALS TO DESIGN PHOTONIC CRYSTAL:

Specify the refractive indices of the materials within your photonic crystal. You can use the predefined material properties for common dielectric materialsor define custom materials.

Material Overview

Material	Selection
Silica glass (mat1)	No domains
Air (mat2)	Domains 1–38

Fig 3.3 Materials used in photonic crystal

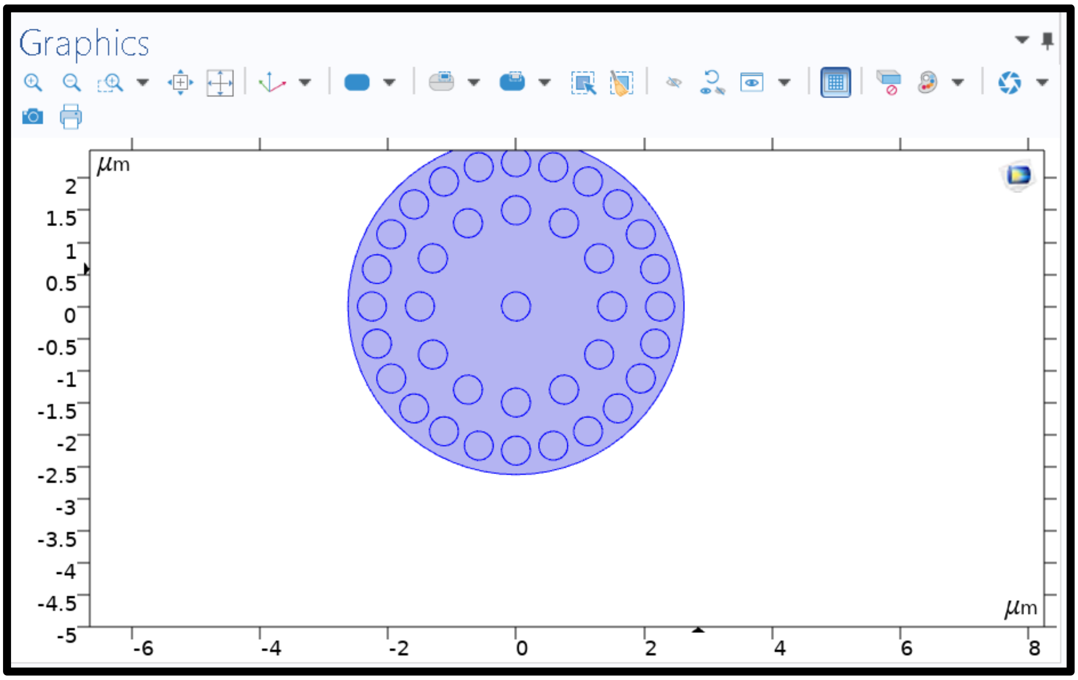

Fig 3.4 Basic structure of photonic crystal

By electromagnetic wave frequency the photonic crystal will be in the abovementioned form.

3.3 MESH AND STUDY:

- After mesh state study state will be applied it contains both modeanalysis and solver configuration.
- Choose an appropriate solver simulation as the frequency domain ortime domain solver, depending on your requirements.
- Create a study to perform simulation and analyze the result. you can sweep the parameters, like wavelength or lattice constant, to study thephotonic bandgap behavior.

RESULT:

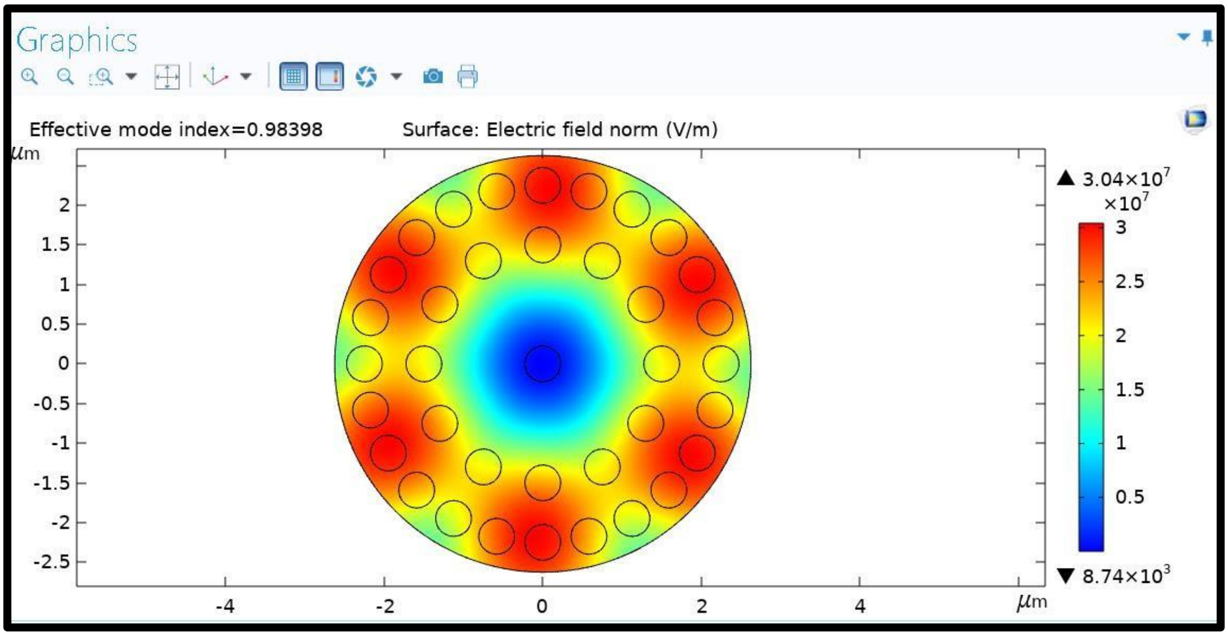

Fig 3.5 3D analysis of photonic crystal(circular wave guide)

- By simulating this photonic crystal we can observe the flow of electricfield and we can also the analyze the effective index mode.
- Based on the simulation results, you may need to iterate on your designand adjust parameters to achieve the desired photonic bandgap or otheroptical properties.
- If necessary, export the data and visualizations for further analysis or presentation.

CHAPTER 4

DESIGN AND SIMULATION OF CAPACITOR USING COMSOL

4.1 CAPACITOR:

- A capacitor with an applied sinusoidally time-varying voltage difference is modeled. A wide frequency range is considered and theimpedance of the device is computed. Solver accuracy is addressed.

- On this Post, I intend to show **how to Design a Parallel-Plate High Voltage Capacitor in Solidworks and Calculate the Capacitance inCOMSOL** according to the specifications of a fictitious Client.

- The same method applied in this study, may be used to design morecomplex design.

- The parametrized design in Solidworks allows one to perform sweepanalysis and optimization analysis in COMSOL, but for this demonstration, only a statonary capacitance study was done.

- A capacitor, in its simplest form, is a two terminal electrical device thatstores electric energy when a voltage difference is applied across the terminals.

- The stored electric energy is proportional to the applied voltage squared andis quantified by the capacitance of the device. This model introduces a model of a simple capacitor, the electric field and device capacitance are solved for under electrostatic conditions.

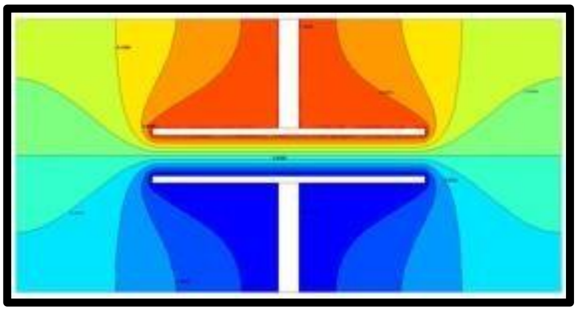

Fig 4.1 capacitor design in COMSOL

4.2 PROCEDURE:

1. Launch COMSOL Multiphysics:
Start COMSOL Multiphysics software on your computer.

2. Create a New Model:
Click on "File" > "New" > "Model" to create a new simulation model.

3. Geometry Definition:
Define the geometry of your capacitive structure. You can create simple geometries within COMSOL or import 3D CAD models if needed. Ensure thatyour geometry represents the physical structure of your capacitor.

4. Physics Setup:

a. Physics Selection:
In the "Model Builder" window, click on "Physics" and select "Electrostatics"or "AC/DC Module" depending on your analysis.

b. Add Materials:
Assign appropriate material properties to your geometry. In the "Materials" node, define the dielectric properties (permittivity) of the material between the capacitor plates. You can choose from the built-in materials or define custom ones.

c. Boundary Conditions:
Set up the boundary conditions for your problem. For a capacitor, you typicallydefine one plate as a fixed potential (e.g., V=0) and the other plate as a free boundary.

5. Meshing:

a. Mesh Generation:
Create a mesh for your geometry by clicking on the "Mesh" node. Ensure thatthe mesh is fine enough to accurately capture the electric field distribution.

6. Study Setup:

a. Parameters:
Define the parameters for your simulation, such as the dimensions of thecapacitor, permittivity, and voltage applied.

b. Solvers:
Choose appropriate solvers for your simulation, depending on whether it's astatic or dynamic analysis.

c. Study Type:
Set up a study type, e.g., a stationary study for static capacitance or a frequency-domain study for AC capacitance.

d.solve:
Click on the "Solve" button to run the simulation. COMSOL will compute theelectric field distribution and other relevant quantities.

7. Post-processing:

a. Capacitance Calculation:

After the simulation is complete, you can calculate the capacitance from the results. You can do this by integrating the electric flux between the capacitor plates and dividing it by the applied voltage:

C = Q / V

Where:

1. C = Capacitance
2. Q = Total charge on one plate (computed from the simulation results)
3. V = Applied voltage

8. Visualize Results:

You can visualize the electric field distribution, charge density, and capacitanceusing COMSOL's post-processing tools and create plots and reports.

9. Optimization (Optional):

If you want to optimize the capacitance for specific design parameters, you canset up optimization studies within COMSOL to find the best configuration.

10. Save and Export:

Save your COMSOL model for future reference and export simulation resultsas needed.

MESH AND STUDY OF A FIXED BEAM CAPACITOR:

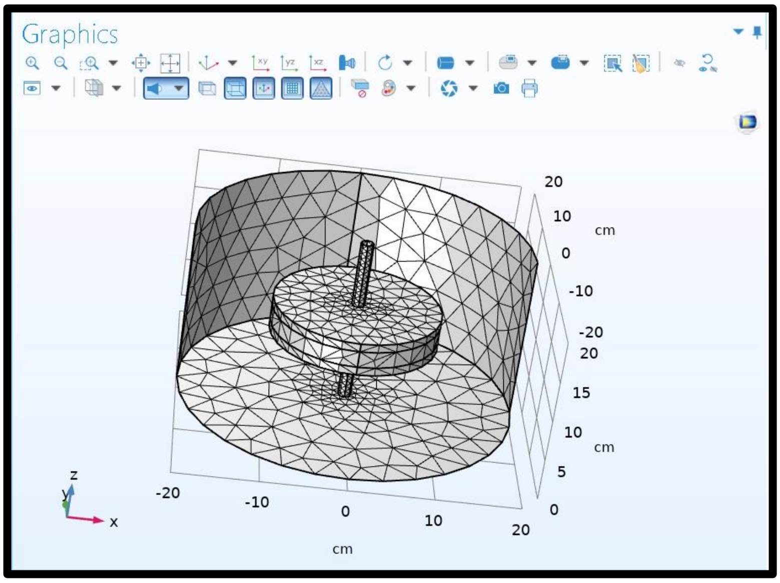

Fig 4.2 Mesh analysis

RESULT AND SIMULATION OF A FIXED BEAM CAPACITOR:

2D PLOT OF THE CAPACITANCE:

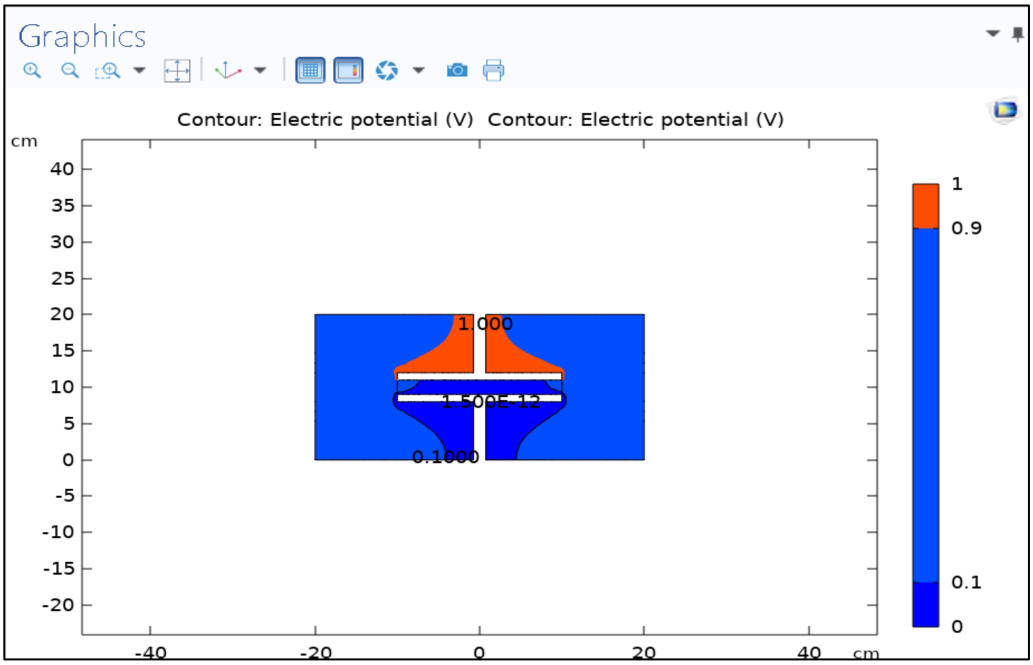

Fig 4.3 2D analysis

3D PLOT OF CAPACITANCE:

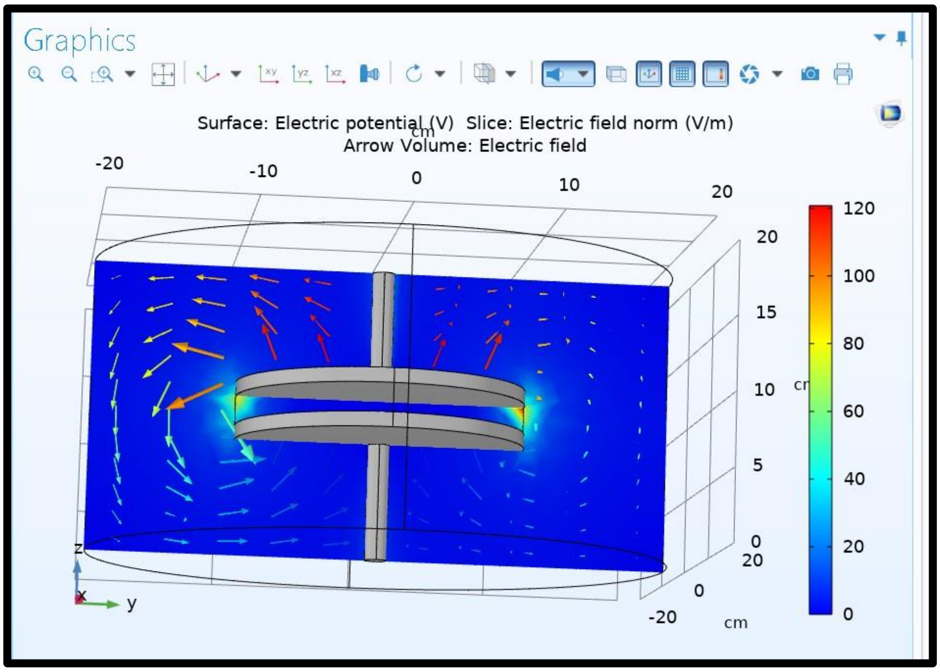

Fig 4.4 3D analysis

CONCLUSION

In conclusion, COMSOL Multiphysics is a powerful and versatile simulation software that enables engineers and scientists to model and solve complex multiphysics problems. Its wide range of application modules, coupled with its user-friendly interface, makes it a valuable tool for research, product development, and optimization across various industries. With its ability to simulate the interaction of multiple physical phenomena, COMSOL Multiphysics plays a crucial role in advancing innovation and solving real-world engineering challenges.

REFERENCE

- Yoke-Choy Leong and S. Weinreb, “Full band waveguide-to-microstrip probe transitions,” 1999 IEEE MTT-S International Microwave Symposium Digest, pp. 1435–1438 vol. 4, 1999.
- “Modeling Transition of Rectangular Waveguide” by Vaibhav Adhikar.
- “Mode Analysis of Electromagnetic Waveguides in Comsol” by Sergey Yankin.
- https://www.comsol.com/wave-optics-module